

AFRL-ML-WP-TP-2007-439

**3-DIMENSIONAL CRACK GROWTH
BEHAVIOR OF TURBINE ENGINE
MATERIALS (Preprint)**

Christine Esperanza



JANUARY 2007

Approved for public release; distribution is unlimited.

STINFO COPY

This work was funded in whole or in part by Department of the Air Force contract F33615-02-2-5800. The U.S. Government has for itself and others acting on its behalf an unlimited, paid-up, nonexclusive, irrevocable worldwide license to use, modify, reproduce, release, perform, display, or disclose the work by or on behalf of the U. S. Government.

**MATERIALS AND MANUFACTURING DIRECTORATE
AIR FORCE RESEARCH LABORATORY
AIR FORCE MATERIEL COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OH 45433-7750**

REPORT DOCUMENTATION PAGE					<i>Form Approved</i> <i>OMB No. 0704-0188</i>	
The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.						
1. REPORT DATE (DD-MM-YY) January 2007		2. REPORT TYPE Journal Article Preprint		3. DATES COVERED (From - To)		
4. TITLE AND SUBTITLE 3-DIMENSIONAL CRACK GROWTH BEHAVIOR OF TURBINE ENGINE MATERIALS (Preprint)				5a. CONTRACT NUMBER F33615-02-2-5800		
				5b. GRANT NUMBER		
				5c. PROGRAM ELEMENT NUMBER 62102F		
6. AUTHOR(S) Christine Esperanza				5d. PROJECT NUMBER 4349		
				5e. TASK NUMBER L1		
				5f. WORK UNIT NUMBER 4349L100		
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Wright State University 3640 Colonel Glenn Highway Dayton, OH 45435				8. PERFORMING ORGANIZATION REPORT NUMBER Project No. 138		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) Materials and Manufacturing Directorate Air Force Research Laboratory Air Force Materiel Command Wright-Patterson AFB, OH 45433-7750				10. SPONSORING/MONITORING AGENCY ACRONYM(S) AFRL-ML-WP		
				11. SPONSORING/MONITORING AGENCY REPORT NUMBER(S) AFRL-ML-WP-TP-2007-439		
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.						
13. SUPPLEMENTARY NOTES PAO Case Number: AFRL/WS 06-1285, 20 March 2007. This work was funded in whole or in part by Department of the Air Force contract F33615-02-2-5800. The U.S. Government has for itself and others acting on its behalf an unlimited, paid-up, nonexclusive, irrevocable worldwide license to use, modify, reproduce, release, perform, display, or disclose the work by or on behalf of the U. S. Government.						
14. ABSTRACT The objective of "3-Dimensional Crack Growth Behavior of Turbine Engine Materials" is to evaluate 2-dimensional and 3-dimensional crack growth and material mechanics in turbine engine components and its corresponding lab specimens under thermo mechanical loading it configurations. In particular, experiments are conducted on titanium and nickel-base super alloys under simulated engine loading conditions. The resulting residual stresses, the deformation and displacement of the geometry, and the initiation and growth of cracks are analyzed. Such experiments are simulated in finite element analysis (FEA) computer programs. Previous geometries analyzed in this project include turbine disk blades, horseshoe-shaped beams, and dog bone specimens.						
15. SUBJECT TERMS						
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT: SAR	18. NUMBER OF PAGES 18	19a. NAME OF RESPONSIBLE PERSON (Monitor) James Blackshire 19b. TELEPHONE NUMBER (Include Area Code) (937) 255-0198	
a. REPORT Unclassified	b. ABSTRACT Unclassified	c. THIS PAGE Unclassified				

3-Dimensional Crack Growth Behavior of Turbine Engine Materials

Project No. 138

**Christine Esperanza
Wright State University**

31 January 2007

**Government Project Leader
Dr. Reji John and Lt Sigfried Haering
AFRL/MLLMN**

**Southwestern Ohio Council for Higher Education
Student Research Program
Agreement Number: F33615-02-2-5800**

Acknowledgments

I would like to thank Dr. Reji John for allowing me this opportunity to work at AFRL. Also, I would like to thank Dennis Buchanan, Sigfried Haering, Chris Fetty, and Bence Bartha for additional assistance in my work. Finally, I would like to thank my fellow students in Room 30, Kyle Bryan, Brian Gockel, and Jeremy Tumpak, for the informative feedback and camaraderie that fellow engineering students can provide.

General Description of Project

The objective of “3-Dimensional Crack Growth Behavior of Turbine Engine Materials” is to evaluate 2-dimensional and 3-dimensional crack growth and material mechanics in turbine engine components and its corresponding lab specimens under thermomechanical loading configurations. In particular, experiments are conducted on titanium and nickel-base superalloys under simulated engine loading conditions. The resulting residual stresses, the deformation and displacement of the geometry, and the initiation and growth of cracks are analyzed. Such experiments are simulated in finite element analysis (FEA) computer programs. Previous geometries analyzed in this project include turbine disk blades, horseshoe-shaped beams, and dog bone specimens.

There are several reasons for using FEA programs. First, it may be desired to garner the results (residual stress, crack growth behavior) from the FEA program and compare the results from the previous laboratory testing or, the given loading conditions may be analyzed in the program and the outcomes may be used to predict the appropriate effects in the lab testing. Another reason a program may be used is to determine the optimum geometry to avoid certain ranges of stresses. Still another need for the analysis programs may be to check the accuracy of one FEA program to another; for example, results may be generated in both ANSYS (Analysis

System) and ABAQUS and compared to theoretical calculations to check which program yields better predictions for a given problem.

Description of Research

Software was the main tool in this project. ANSYS and ABAQUS are programs that can model and analyze both 2D and 3D problems. Since ANSYS and ABAQUS have these capabilities, functions such as modeling and meshing a given geometry, adding appropriate material properties, creating analogous boundary and loading conditions and analyzing the problem are all possible with both programs. For crack growth, FRANC3D is used to insert a crack and re-mesh the present mesh in ANSYS or ABAQUS; FRANC3D can also be used to extend existing cracks and create a new mesh. Then the mesh, with a crack block included, is analyzed in either ANSYS or ABAQUS. In conjunction with ANSYS or ABAQUS, FRANC3D is also used to compute stress intensity factors.

A typical methodology is as follows. A specimen's geometry is modeled and meshed in the appropriate program. Then material properties, such as Young's Modulus and Poisson's Ratio, are entered into the program. The next step is to model the loading conditions under which the specimen will be exposed to. For example, typical loading of a test specimen model would be axial loading on both ends with rigid body motion constraints mimicking grips of a test frame. Usually, the problem will first be analyzed through these steps before continuing to more complex analyses, such as crack deformation; this is to ensure that the FEA program is outputting appropriate results (values close to expected theoretical calculations). Once the problem is run successfully through an FEA program, results are output.

The current focus of analysis is the modeling and analysis of a helicopter lift frame specimen; see Figures 1 and 2 for a one-quarter model of the lift frame. The material is

Aluminum Alloy 7010 T73651. The loads are applied at each bolt hole at each end of the specimen, which should result in a nominal stress of about 1 megapascal (MPa). Further analysis will insert a crack along the center hole of the specimen. The purpose is to compare results published in the paper “Life prediction from high cycle dynamic components using damage tolerance and small threshold cracks” by Vaughan and Chang.

In ANSYS, a quarter-model was built. A one-fourth geometry helps lower the analysis time for ANSYS. Symmetry conditions can be applied later to the planes of symmetry to alert the program that the present model is one-fourth of a total geometry. The following figures show two different views of the model.

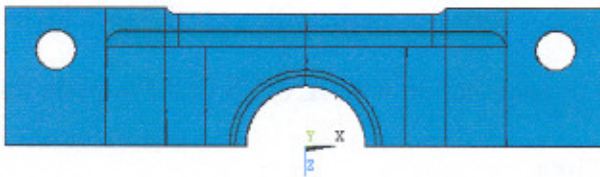


Figure 1. This is a top view of the specimen in ANSYS.

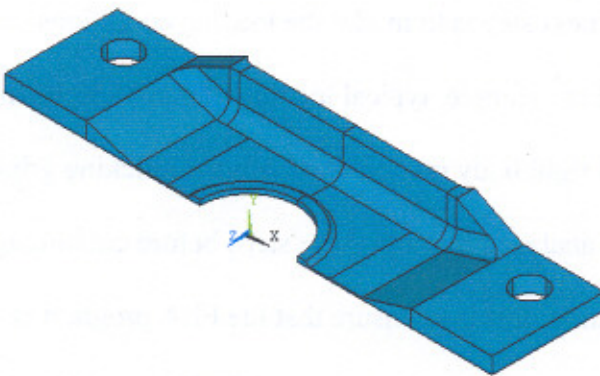


Figure 2. This is an isometric view of the specimen in ABAQUS.

After the geometry was built, the material properties for Aluminum Alloy 7010 T73651 had to be entered in the “Materials Models” menu in ANSYS. All model dimensions in millimeters. The Elastic Modulus is entered as $720,000 \text{ N/mm}^2$; Poisson’s ratio is 0.33.

A mesh had to be built next. Figure 3 shows where hexahedral and tetrahedral element types were located in the model.

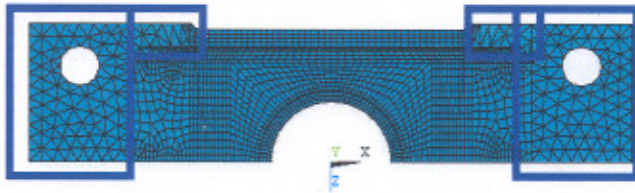


Figure 3. The model after a mesh has been added. The boxed areas indicate where the elements are tetrahedral; the remaining parts of the mesh are hexahedral.

Next the boundary and symmetry conditions were added. The middle line of the specimen was restricted in the X direction in order to keep the middle part of the specimen fixed; see Figures 4 and 5. Symmetry conditions were added to the bottom plane and to the side plane as indicated in Figures 6, 7, and 8.

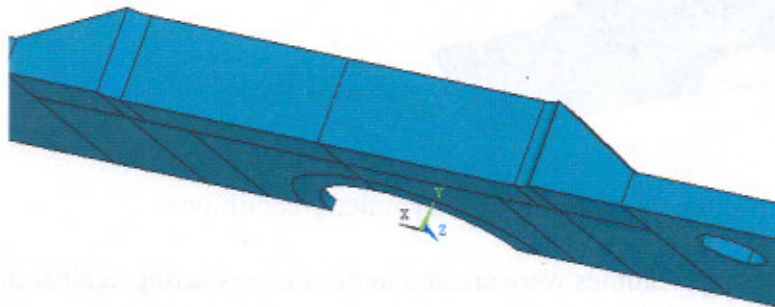


Figure 4. The line, indicated by the three arrows, is where the geometry is constrained in the x-direction.

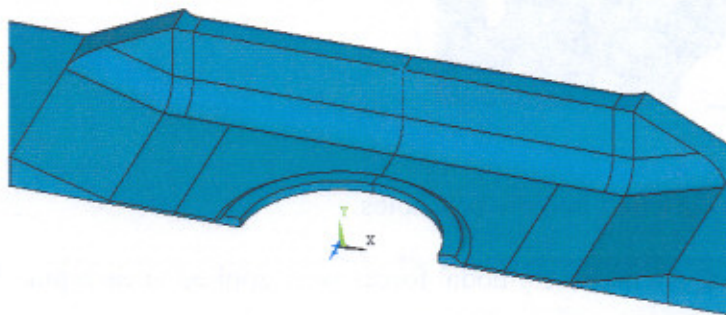


Figure 5. This is a front view to show where the constrained line is in relation to the rest of the model.



Figure 6. The symmetry boundary condition was applied to the bottom and side planes, as indicated here by the boxes.

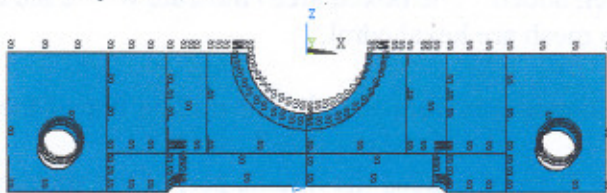


Figure 7. A view of the bottom plane of symmetry.

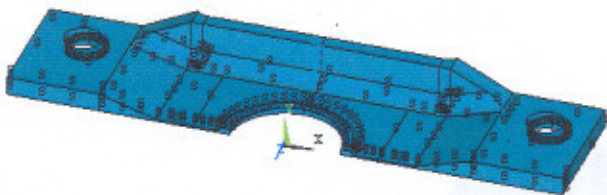


Figure 8. An isometric view to show the boundary and symmetry conditions.

To add loads, two additional volumes were created to mimic pins acting against the bolts holes. Figure 9 shows the model with the “pins” included. They were also meshed.

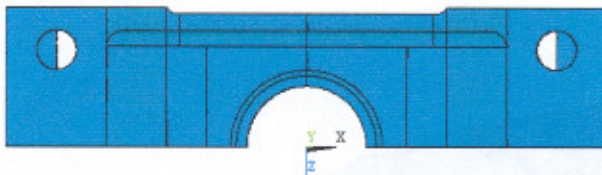


Figure 9. The model with the pins inserted into the bolt holes.

Once the pins were meshed, the necessary nodal forces were applied to each pin. Load values such that the resulting stress in the cross sectional areas of the ends of the geometry were

1 MPa. This is calculated to be 690 N at each bolt hole. Since three nodes were acted on, each node underwent a force of 230 N, shown in Figure 10.

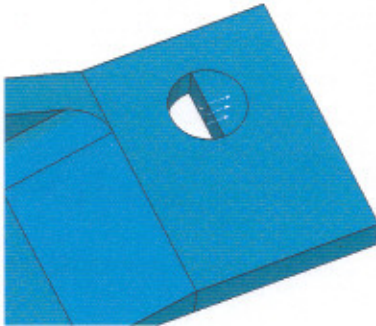


Figure 10. One of the pins showing where the node forces were applied.

After all of the boundary, symmetry, and loading conditions were applied, the geometry was solved using ANSYS.

Results

In the Post-Processing menu, the x-component of stress was the main result of interest. This is because, along the middle plane, it is the hoop stress for the main hole in the model. This stress will cause the future crack to open. Figure 11 shows the contour plot.

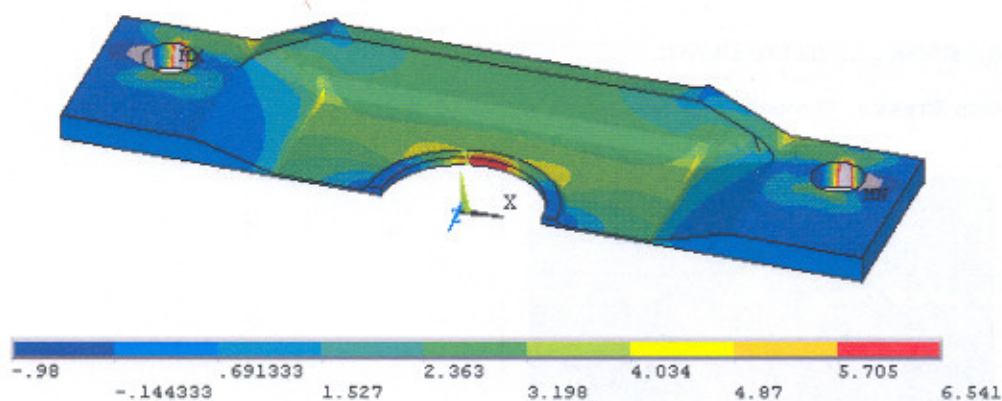


Figure 11. The contour plot for X stress after the part has been analyzed in ANSYS.

Hoop stress was extracted from along the thickness of the specimen as well as along the radius; this is where the crack will be inserted later. See Figures 12 and 13 to follow where hoop stress was extracted.

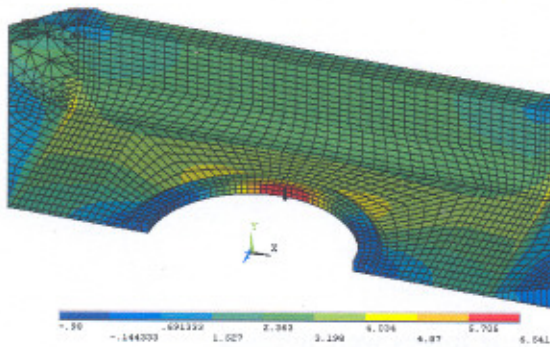


Figure 12. One path of interest was along the thickness of the main hole (the thick black line), along its line of symmetry. This path is referred to as the “Mid Thickness.”

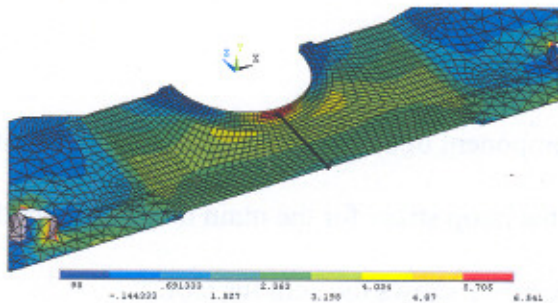


Figure 13. This is the other path of interest in the model. It is along the bottom plane, along the center line. It is known as the “Radial Depth.”

The resulting stress graphs are shown.

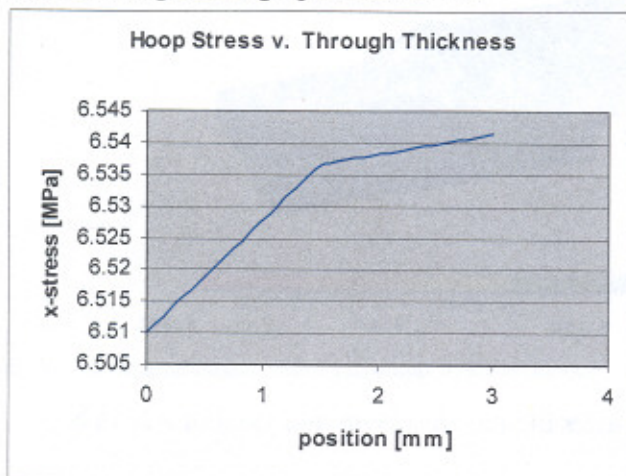


Figure 14. The graph of hoop stress plotted against the through thickness.

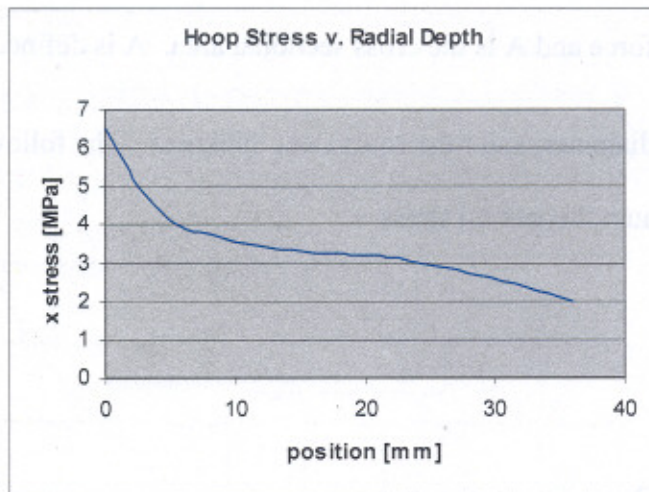


Figure 15. The graph of hoop stress plotted against the radial depth.

To check how the ANSYS results for the highest hoop stress compared to theoretical values, the theoretical stress concentration factor were used. Specifically, Table E-15, Figure E-15-1 from the book, *Mechanical Engineering Design*, was referenced.

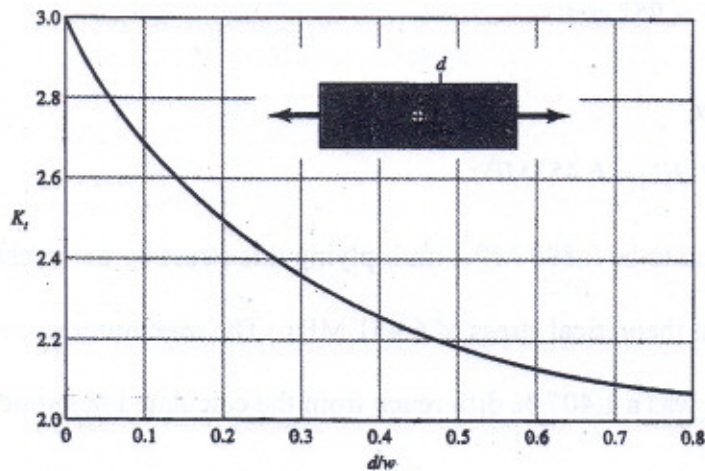


Figure 16. This is Figure E-15-1 from Table E-15 in *Mechanical Engineering Design*.

The dimensions at the cross section of the lift frame are as follows: d (diameter) = 60mm and w (width) = 132mm. From these dimensions, it was found from Figure E-15-1 that the stress concentration factor, K_t^* , was about 2.225. To find the highest stress that occurs in the cross section, K_t^* is multiplied by the net sectional stress. The net sectional stress is found by the

formula $\sigma_0 = \frac{F}{A}$, where F is the axial force and A is the cross sectional area. A is defined by the

formula $A = (w-d) t$, w being width, d diameter, and t the equivalent thickness. The following is a sample calculation to find the maximum theoretical stress.

$$\sigma_{0,max} = K_t \times \sigma_0$$

$$\sigma_0 = \frac{F}{A}$$

$F = \text{axial force}$

$A = (w-d) * t$,

$w = \text{width} = 132\text{mm}$

$d = \text{diameter} = 60\text{mm}$

$t = \text{equivalent thickness (due to the three different thicknesses)}$

	thickness	% width	t*%
sides	50mm	0.222222	11.11111
majority	2mm	0.638889	1.277778
hole lip	6mm	0.138889	0.833333

From this table, $t = 13.222\text{mm}$

Then, $A = (132-60) * (13.222) = 952\text{mm}^2$

$F = 2760\text{N}$

$$\sigma_0 = \frac{F}{A} = \frac{2760\text{N}}{952\text{mm}^2} = 2.899\text{MPa}$$

$$\sigma_{0,max} = K_t \times \sigma_0 = 2.225 \times 2.899\text{MPa} = 6.451\text{MPa}$$

The net sectional stress was found to be 2.899 MPa; multiplying this stress by the stress concentration factor gave us a maximum theoretical stress of 6.451 MPa. The maximum stress found in ANSYS was 6.541 MPa. This was a 1.407 % difference from the calculated theoretical value.

Due to the small percentage difference between theoretical and analytical maximum stress, it appears that there is agreement between the two results. The next step in this project will be to create a crack block mesh and re-mesh the entire geometry using FRANC3D. Because working involving crack analysis is currently ongoing, it is not included in this current technical report.

Works Cited

Shigley, Joseph, Mischke, Charles, and Budynas, Richard. Mechanical Engineering Design.

6th ed. New York: McGraw-Hill Science/Engineering/Math, 2001.